



# YAMS A fully Automatic Adaptive Isotropic Surface Remeshing Procedure

Pascal Frey

## ► To cite this version:

Pascal Frey. YAMS A fully Automatic Adaptive Isotropic Surface Remeshing Procedure. RT-0252, INRIA. 2001, pp.36. inria-00069922

**HAL Id: inria-00069922**

**<https://inria.hal.science/inria-00069922>**

Submitted on 19 May 2006

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

YAMS

*A fully automatic adaptive isotropic  
surface remeshing procedure*

Pascal J. FREY

**No 0252**

November 8, 2001

\_\_\_\_\_ THÈME 4 \_\_\_\_\_

 *rapport  
technique*



YAMS  
**A fully automatic adaptive isotropic  
surface remeshing procedure**

Pascal J. FREY

Thème 4 — Simulation et optimisation  
de systèmes complexes  
Projet Gamma

Rapport technique n°0252 — November 8, 2001 — 36 pages

**Abstract:**

This technical note describes the main features of YAMS\*, an automatic adaptive surface remeshing tool. The aim of the software, the input and output files and the list of error messages are defined in this document. A number of typical application examples are provided to explain the various possibilities of the code. Finally, a short technical description of the algorithm and the various ways to obtain the software are given.

YAMS has been developed within the GAMMA research project at INRIA-Rocquencourt.

This document describes the features of the current version :  
release V2 . 1 (June, 2001).

**Key-words:** surface mesh, decimation, mesh adaptation

*(Résumé : tsvp)*

\* This software was registered with the APP under n° IDDN.FR.001.410006.00.R.P. 1999.000.20600 on october 4, 1999.

# YAMS

## Résumé :

Ce rapport technique décrit les principales fonctions de YAMS<sup>†</sup>, un logiciel automatique de remaillage adaptatif de surfaces. Les fonctionnalités du logiciel, les formats des fichiers d'entrée/sortie et une liste de messages d'erreur sont décrits dans ce document. Des exemples d'applications sont fournis pour expliquer les différentes possibilités offertes par l'algorithme et les moyens de se procurer le code.

YAMS a été développé au sein du projet GAMMA à l'INRIA-Rocquencourt.

Ce document décrit les fonctionnalités de la version courante : version V2 . 1 (Juin 2001).

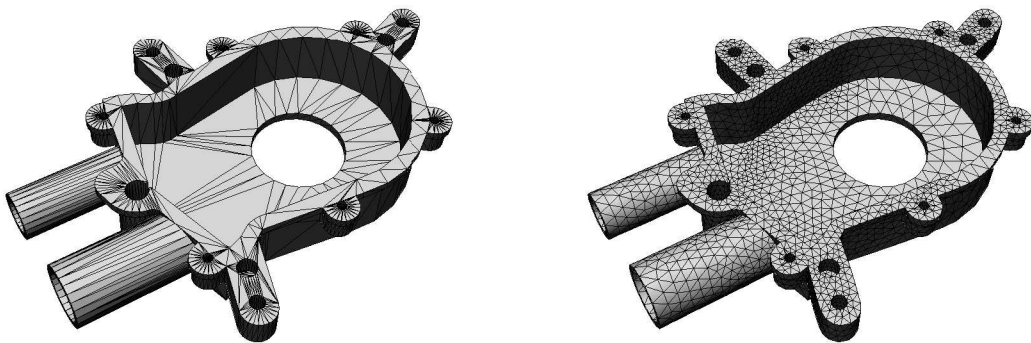


Figure 1: *Exemple de remaillage de surface sur une pièce mécanique.*

**Mots-clé :** maillage de surface, décimation, adaptation de maillage

<sup>†</sup> Ce logiciel a été enregistré à l'APP sous le n° IDDN.FR.001.410006.00.R.P. 1999.000.20600 le 4 octobre 1999.

# Contents

<b>1</b>	<b>Surface remeshing procedure</b>	<b>5</b>
1.1	Context . . . . .	5
1.2	Yams overview . . . . .	5
<b>2</b>	<b>Input data</b>	<b>6</b>
2.1	Geometry description . . . . .	6
2.2	Metric specification . . . . .	6
2.3	Software integration . . . . .	6
<b>3</b>	<b>Output and Errors</b>	<b>7</b>
3.1	Output files . . . . .	7
3.2	Error messages . . . . .	7
<b>4</b>	<b>Technical information</b>	<b>8</b>
4.1	Language, platforms . . . . .	8
4.2	Memory requirement . . . . .	8
4.3	CPU time estimation . . . . .	8
4.4	Distribution . . . . .	8
<b>5</b>	<b>How to use YAMS</b>	<b>10</b>
5.1	Flowchart of the algorithm . . . . .	10
5.2	Command line and options . . . . .	10
5.2.1	Command line . . . . .	10
5.2.2	Other options . . . . .	11
5.3	Control parameters . . . . .	11
5.4	Physical attributes . . . . .	12
<b>6</b>	<b>Application examples</b>	<b>13</b>
6.1	Surface simplification . . . . .	13
6.1.1	High-quality decimation . . . . .	13
6.1.2	Geometry-driven decimation . . . . .	15
6.1.3	Application : creation of hierarchical meshes . . . . .	17
6.2	Surface mesh enrichment . . . . .	18
6.2.1	High-quality geometric surface meshes . . . . .	18
6.2.2	Geometric surface meshes . . . . .	19
6.2.3	Mesh gradation control . . . . .	19
6.3	Surface mesh optimization . . . . .	21
6.3.1	Mesh quality improvement . . . . .	21
6.3.2	Surface approximation improvement . . . . .	21
6.4	Mesh adaptation . . . . .	23
6.5	2D applications . . . . .	25
<b>7</b>	<b>Limitations</b>	<b>26</b>
7.1	Defining the surface geometry . . . . .	26
7.2	CAD repair tool . . . . .	27
7.3	About corners and ridges . . . . .	27

<b>8</b>	<b>Appendix</b>	<b>28</b>
8.1	List of error messages . . . . .	28
8.2	Isotropic remeshing . . . . .	30
8.2.1	Shape quality . . . . .	30
8.2.2	Size quality . . . . .	30
8.2.3	Geometric support . . . . .	30
8.3	File formats . . . . .	31
8.4	The mesh format . . . . .	31
8.4.1	Example . . . . .	32
8.5	The msh2 format . . . . .	33
8.6	the bb format . . . . .	33
8.7	List of FAQ's . . . . .	34
	<b>References</b>	<b>36</b>

# 1 Surface remeshing procedure

## 1.1 Context

Usually, a surface remeshing algorithm is a procedure to transform an arbitrary triangulation (eventually containing ill-shaped triangles) into a high-quality mesh. This procedure is especially useful to generate surface meshes in view of numerical simulations or for high-end visualization purposes. It can also be used to reduce the amount of data required to represent a complex 3D model. In all cases, the resulting mesh should correspond to a good and accurate (piecewise linear) surface approximation with a reasonable number of elements. Hence, the element size is locally related to the surface curvature. In this respect, such a mesh can be qualified as "nearly optimal".

## 1.2 Yams overview

YAMS is a *fully automatic adaptive surface remeshing tool* that takes any arbitrary surface triangulation and produces an adapted (curvature-based) surface mesh. The remeshing procedure is governed by the geometric surface properties (a *geometric size map* that relates the element size to the local curvature is defined) as well as by a *metric map* (for instance provided by an *a posteriori* error estimate), both being defined at the vertices of the input mesh. This scalar map<sup>1</sup> provides a suitable element size at each mesh vertex to locally discretize the surface accurately. This size map is sufficient to control the mesh density and the mesh gradation as well as the element shape quality.

The initial surface description can be any triangulation, for instance supplied by a CAD-CAM package or any surface reconstruction algorithm. The input data needs to be *orientable* and must be composed of triangles and/or quadrilaterals only. Non-manifold models are handled.

**Remark 1.1** *In addition to these requirements, additional entities and information can be supplied to help remeshing the surface (constrained entities, corners, ridges, normals, etc.).*

The algorithm proceeds in three steps. At first, the surface is analyzed and its intrinsic properties are computed based on the actual discrete representation. This results in the construction of the geometric size map. Then, the triangulation is processed and modified according to the size prescription. To this end, topological and geometrical operations are performed to achieve a mesh in which the element size and the element shape quality are controlled. Finally, a local optimization stage is carried out to improve the element shape quality.

YAMS also produces diagnostics about the correctness of the surface (regarding its orientation) and various histograms (regarding the quality of the approximation).

---

<sup>1</sup>It is a scalar map if the mesh is to be isotropic.



## 2 Input data

The geometry of the surface is described using a simple mesh format. The specification of a metric map in the context of mesh adaptatation is also described hereafter.

### 2.1 Geometry description

The surface geometry is described preferably using the mesh file format. (see Appendix for the complete description of this format). For compatibility purposes the "old" msh2 format is also supported in this release. Notice that if quads are supplied, they are automatically converted to triangles.

The mesh format allows to precisely describe the surface features. For instance, corners, ridges, normals can be supplied in this file format to provide more accurate information to the remeshing procedure. This is the only definition of the surface that the program will know as there is no link with a CAD modelling system.

- The mesh format.  
This format is composed of a single (ASCII or binary) file, `xxx.mesh` or `xxx.meshb`. This file contains all the information needed to describe entirely the surface mesh and the underlying geometry. It is organized as a series of fields, identified by keywords. In addition to the vertex coordinates and the list of faces, it allows one to specify additional information such as corners, edges, ridges, required entities (that must be preserved in the final mesh), etc.
- The msh2 format.  
This very crude format is composed of two ASCII files, `xxx.points` and `xxx.faces` describing the geometry (vertex coordinates) and the topology (list of elements) of the surface mesh, respectively.

### 2.2 Metric specification

A user supplied size map can be described as a single data file, `xxx.bb`, containing scalar values (*i.e.*, a sizing information) which are associated with the vertices (or with the elements) of the given surface mesh. By default, when reading an input file (`test.mesh`, for instance), the program will look to see if there is any size specification attached to this file (`test.bb`). In this case, the user-specified metric will be combined with the intrinsic geometric metric in order to preserve the element size in highly curved region while modifying the standard size by the one supplied by the user in the planar regions.

### 2.3 Software integration

If YAMS is integrated in a software package, only the input and output routines need to be modified, for efficiency and compatibility purposes. In this context, no more than 4 routines need to be modified and adapted to support the user file formats.

## 3 Output and Errors

### 3.1 Output files

Usually, a single output file is produced at completion, representing the resulting mesh. This file, `xxx.meshb`, is written in the binary mesh format by default. It contains the geometry and topology of the piecewise linear approximation (*i.e.*, the vertex coordinates and triangles) as well as additional data relative to the specific features : the corners and ridges, for instance. A more complete file can also be exported in mesh format containing normals, tangents, etc, if desired.

If specified, the current size map can be written in a `xxx.bb` file (similar to that used as input file).

Notice that the output files are fully compatible with the input files, thus making it easy to use YAMS as an optimizer (the output files becoming the input files when running the code again).

### 3.2 Error messages

At completion of the remeshing procedure, two cases can be encountered :

- A mesh file has been created, hence meaning that the resulting mesh is valid. However, this does not necessarily means that the mesh is a very accurate surface approximation, as this result is strongly related to the parameter specification (see Section 5.2).
- No output file has been produced : either the data are incorrect (or missing), or the algorithm detected a blocking error and stopped, writing an error code (see below).

The program may stop for various reasons, for instance :

- the input file (in the `msh2` format) contains faces other than triangles or quadrilaterals,
- the allocated memory size is not sufficient. The solution is then to increase the memory size : option `[-m MegaBytes]`,
- the surface is not *orientable* (a Möbius strip or a Klein bottle, for instance).

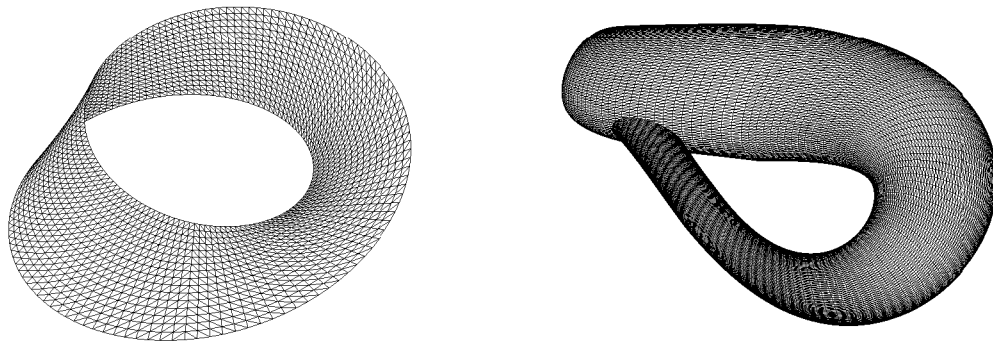


Figure 2: Examples of non-orientable surface meshes : a Möbius strip (left-hand side) and a Klein bottle (right-hand side).

## 4 Technical information

### 4.1 Language, platforms

From the beginning, the program has been entirely written in C (ANSI). The current version consists of about 17,000 lines of optimized source code. Hence, the code is very portable and has been successfully compiled and tested on all major computer architectures (*i.e.*, HP, Sun, SGI, IBM, Intel-based PC, etc.) and operating systems (Unix/Linux, WindowsNT, Mac OS).

### 4.2 Memory requirement

The program requires roughly about 350 bytes per vertex thus making it possible to estimate *a priori* the required memory size to complete a job. As an example of memory space needed, a mesh of 543,652 points (1,087,716 triangles) requires 182 MegaBytes (*i.e.*, 351 bytes per point). Conversely, a workstation having 512 MegaBytes of memory can handle a mesh of about 2,750,000 points.

The memory is dynamically allocated at the data file reading stage, also, it obeys the following simple rule. If the user choses to only simplify or optimize the given mesh, then the memory allocated is strictly the memory required to store the input mesh. However if the user selects a mesh enrichment option, the memory is automatically set up to 1.5 times the initial size (*i.e.*, required to store the input mesh). This simple bound prevents any memory overflow because of small artifacts in the initial surface discretization. If during the execution, the memory allocation becomes insufficient, the following error message is issued :

```
ERR 4002, proc, UNABLE TO CREATE NEW TRIANGLE
```

In such a case, the user can either decide to run the program again and specify the total amount of memory using the `[-m MegaBytes]` option (see below) or chose to change the minimal size prescribed in the configuration file.

### 4.3 CPU time estimation

The CPU time required to obtain the final mesh (including the I/O) is obviously related to the complexity of the surface, as well as to the parameters and options specified. Nevertheless, one can reasonably expect (see table in section 6) :

- a few seconds to simplify a mesh of several thousands of elements and
- a few minutes to enrich a mesh of several hundreds of thousands of elements.

### 4.4 Distribution

An evaluation copy of YAMS software for a limited period of time can be obtained by contacting the authors at INRIA :

Pascal FREY  
 INRIA, Domaine de Voluceau  
 BP 105, 78153 Le Chesnay cedex, France  
 Email: [pascal.frey@inria.fr](mailto:pascal.frey@inria.fr)  
<http://www-rocq.inria.fr/gamma/yams/yams.html>

or by contacting Simulog, the distributor of YAMS worldwide :

SIMULOG

1, rue James Joule

78286 GUYANCOURT cedex, France

Fax number: (33) 1 3012 2727 (attention Mark Lorient)

Email: [mark.lorient@simulog.fr](mailto:mark.lorient@simulog.fr)

<http://www.simulog.fr>,

section Compos. maillages/Meshing Software Components

Simulog provides State-of-the-Art meshing software components that ensure software developers to successfully create their CAD/CAE applications. In this respect, Simulog can deliver source code for integration purposes or binary files (on various platforms) of the current version of YAMS.

## 5 How to use YAMS

In this section, indications about the general scheme of the algorithm are provided as well as some hints to achieve specific results.

### 5.1 Flowchart of the algorithm

As indicated above, the mesh algorithm is based on local topological and geometrical modifications (see reference [4] for more details). The general scheme of the method includes the following steps :

- reading the input file(s), and the metric file (if any),
- surface analysis (orientation, normal computation, ridge and corners identification, etc.),
- iterative local modifications using geometrical operators (node repositioning, vertex insertion/deletion) and topological operators (edge flipping),
- mesh optimization (by relocating the vertices or using topological operators to improve the element aspect ratio while conforming to the sizing map),
- writing the output file(s).

### 5.2 Command line and options

YAMS is invoked via a single command line. At this level, only a few options can be specified.

#### 5.2.1 Command line

The usual way of starting YAMS is to type in the following line :

```
yams [-O n] [InputFile].
```

Also, typing : `yams -h` will give the meaning of all the various options.

By default, the name of the output file corresponds to the basename of the input file followed by the extension `.d` (for example, `test.d.meshb` if the input is `test.meshb`). The user can supply any file name for the output file.

The user needs to specify which treatment has to be carried out on the input data : *i.e.*, simplification, enrichment, optimization, adaptation or smoothing. The meshing options are specify using the command : `yams -O n`, where the parameter `n` means :

- 0 optimization only (quality improvement). The number of nodes remains identical to that of the initial surface triangulation.
- 1 high-quality surface remeshing. The final mesh is suitable for numerical simulations (nearly equilateral triangles, mesh density related to the local curvature).
- 2 geometric surface remeshing (Hausdorff based). The distance between the final mesh and the original one is bounded by a threshold value. The triangles are slightly anisotropic.
- 3 pattern-based geometric surface enrichment. Nodes are added for geometric purposes (*i.e.*, improving the geometric approximation).
- 9 improve surface approximation. This is useful for processing "Marching-Cubes" like meshes (e.g., to remove staircase effects).

Notice that **negative** values of  $n$  for options 1 and 2 can be specified, thus indicating that the mesh needs only to be simplified (this option accelerates the global processing of the mesh). New nodes are created on an internally build  $G^1$  support to improve the quality of the geometric approximation.

### 5.2.2 Other options

Other options can also be specified on the command line :

- v [n]      Verbosity level (to get more information during the process),
- b          Save .bb metric file,
- c [n]      Save connected component  $n$  only,
- e          Save extended information in output file (normals, tangents, etc.),
- f          Save formatted (ascii) output file,
- m [n]      Set the memory size to  $n$  MegaBytes,
- nm        Create point on straight edge (no mapping on geometric support),
- no        Do not write output file,
- ns        Disable node smoothing during the optimization stage,
- vrml      Save VRML (level 1) file.

**Remark 5.1** *By default, only the option -O [n] and the name of the data file need to be specified.*

## 5.3 Control parameters

In addition to the previous options, a few control parameters are required to properly characterize the desired surface approximation<sup>2</sup>. These parameters are stored in a simple text file, named `xxx.yams` (associated with `xxx.mesh` file) or `DEFAULT.yams`. The file structure is organized as a series of fields, composed of a keyword and a scalar value.

**Remark 5.2** *For the sake of simplicity, all parameters have been assigned reasonable default values, thus making it possible to run YAMS without creating such a file.*

Parameters that can be specified are as follows (notice that all keywords are **not** case sensitive) :

- **Absolute**  
indicates that the tolerance and size parameters are specified in model unit. Default is to consider relative values (e.g. in 1/1000 of the bounding box size).

<sup>2</sup>This make surface remeshing more tedious to control than volume meshing !

- **Tolerance,  $\delta$**  (Float)  
used to specify the maximum allowable chordal deviation  $\delta$ .  
Default value is 1/1000 of the bounding box size (or model units).
- **GeomApp,  $\varepsilon$**  (Float)  
used to control the surface smoothness (bound the maximal deviation of the mesh edges from the tangent plane at mesh vertices).  
Range : 0.001 to 0.1, default value : 0.01.
- **Ridge,  $\alpha$**  (Float)  
allows to specify the angular value for the detection of ridges and corners (*i.e.*, the measure of the angle between the normals of adjacent faces or edges).  
The default value is 45 degrees.
- **Gradation,  $grad$**  (Float)  
indicates the rate of the size variation between neighboring elements.  
Values : -1 if no gradation is desired, range : 1. to 100, default : 1.3.
- **MinSize,  $h_{min}$**  (Float) and **MaxSize,  $h_{max}$**  (Float)  
allow to prescribe a minimal (resp. maximal) element size.  
Default  $h_{min}$  is 5/1000 of the bounding box size (or model units). No default value for  $h_{max}$ .
- **NbIter,  $n$**  (Integer)  
used to prescribe the number of smoothing iterations when invoking the option `-O 9`. The default value is set to 50.

## 5.4 Physical attributes

Inherently, the surface remeshing algorithm preserves the physical attributes specified in the input file (vertex, edge and face references). However, as can be easily understood, the numbering of the vertices is affected by the algorithm. However, with option `-O 0`, it may be possible to preserve vertex numbers (although not the triangle numbering).

The different surface connected components are automatically identified based on the ridge detection. In the case where several components have been identified, the algorithm can retain only one of them (using the option `[-c n]` in the command line).

## 6 Application examples

This aim of this large section is to emphasize the various features of the software package YAMS. To this end, we will illustrate the two possible approaches for surface simplification. Then, we will focuss on surface enrichment and illustrate how to obtain a high-quality surface mesh for numerical simulations. Afterwards, surface mesh optimization will be briefly discussed and finally, examples of mesh adaptation will be provided to show the efficiency of the algorithm in the context of numerical simulations based on finite element/volume methods.

### 6.1 Surface simplification

One of the most attractive feature of YAMS is the possibility of reducing the number of mesh entities while controlling :

- i) the quality of the geometric approximation,
- ii) the surface properties (curvatures, ridges, etc.) and
- iii) the global mesh quality.

Numerous fields of applications are concerned with this feature, including computer graphics, cartography, virtual reality, numerical simulation, etc.

Given a surface triangulation, two options of mesh simplification are proposed in the software package, allowing to obtain :

- high-quality (finite element) surface meshes (meshes that are primarily intended for numerical simulations) or
- geometric surface meshes (suitable for visualization purposes in which element shape quality control is less crucial).

#### 6.1.1 High-quality decimation

This possibility corresponds to the option `-O -1` of the command line. The original surface triangulation is simplified according to the geometric size map computed internally. Typically, this option triggers the construction of a high-quality geometric mesh (*i.e.*, a good geometric approximation of the surface). In this kind of mesh, the element sizes are directly related to the local curvature. This option involves three parameters : the tolerance `Tolerance`, the geometric approximation `GeomApp` and the mesh gradation `Gradation`.

Table 1 reports some results for this type of mesh simplification procedure. In this table,  $np$  and  $ne$  represent the number of points and triangles,  $Q_{max}$  and  $Q_{avg}$  represent respectively the worst and average element shape quality (see Appendix),  $Q_{[12]}$  is the percentage of elements having a quality better than 2,  $l_{avg}$  and  $\tau$  represent the average edge lengths and the efficiency coefficient,  $cpu$  denotes the CPU time (on a 550 Mhz HP 9000 workstation). In this table, the first line of each example indicates the initial values and the second line the final results. All examples have been computed using the default parameter values (*i.e.*, no `.yams` file used).

Notice that the average quality is usually very close to 1 and the number of elements having a quality better than 2 is above 90%, although there is no guarantee (nor a numerical bound) on the final mesh quality. In some cases, a few badly-shaped triangles may remain because of some geometrical constraint (e.g., a skinny triangle located between two ridges).



<i>Test cases</i>	$np$	$ne$	$Q_{max}$	$Q_{avg}$	$Q_{[12]}$	$l_{avg}$	$\tau$	<i>cpu (sec.)</i>
voiture	4,557	9,110	15.7	1.83	75.8			-
	3,385	6,786	5.8	1.45	91.85	1.06	0.92	2.6
hotrod5	17,910	34,950	12.7	1.86	71.09			-
	4,779	9,152	12.2	1.37	96	0.84	0.93	6.5
head	67,106	134,208	2.7	1.2	99.98			-
	10,178	20,352	2.7	1.23	99.88	0.77	0.89	19.8
total (Fig. 3, top)	116,384	232,776	$\infty$	3.14	96			-
	25,181	50,370	221	1.4	97.6	0.88	0.91	51.6
igea (Fig. 3, bottom)	134,345	268,686	47.14	1.44	99.08			-
	27,871	55,738	47.14	1.25	99.69	0.72	0.87	59.8
happy	543,652	1,087,716	115	3.42	69.5			-
	173,769	347,950	183.	1.63	90.23	0.43	0.6	295

Table 1: *Statistics relative to high-quality mesh simplification (option -O -1).*

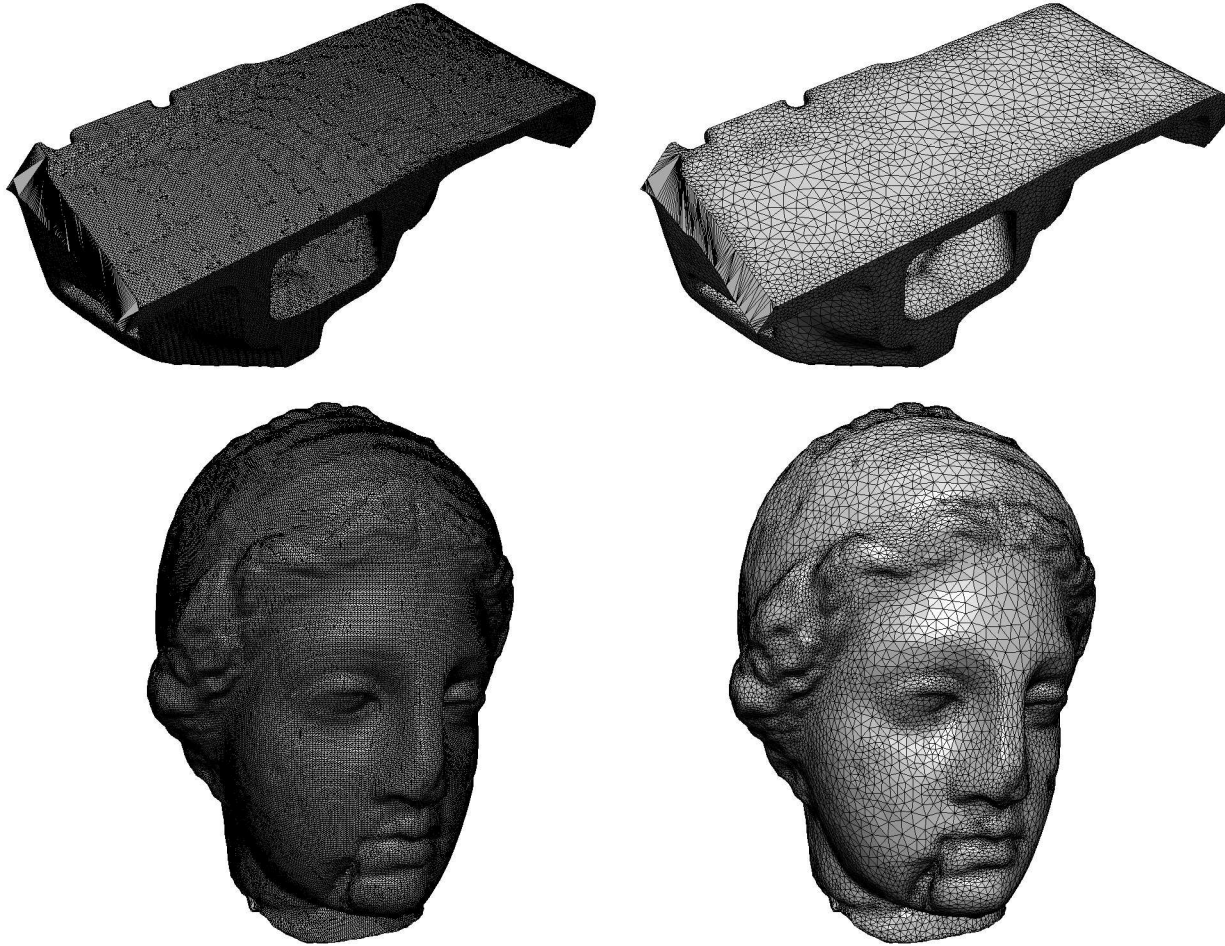


Figure 3: *Examples of high-quality mesh simplification using option -O -1. Left ; original surface meshes. Right : high-quality simplified meshes.*

### 6.1.2 Geometry-driven decimation

This feature corresponds to option `-O -2` of the command line options. It allows the user to generate a surface approximation corresponding to a bounded deviation from the original surface triangulation. The tolerance value can be seen as a *protective envelope* surrounding the original triangulation. In this option, the Hausdorff between the two surfaces is computed to make sure that the surface deviation remains bounded below the threshold value. Usually, the resulting meshes present anisotropic features, the elements being stretched along the main directions of curvature. Hence, resulting meshes are mainly intended for visualization purposes, where the geometric accuracy is needed while the element shape quality is less important. Specifying a `MaxSize` allows the user to bound the natural stretching of the elements.

Table 2 reports some results for this mesh simplification procedure ( $Q_{max}$  represents the worst mesh quality, *cpu* denotes the CPU time on a HP 9000 workstation). In this table, the first (resp. second) line of each example corresponds to the initial (resp. final) meshes.

<i>Test cases</i>	<i>np</i>	<i>ne</i>	$Q_{max}$	<i>cpu</i> (sec.)
pda06 (Fig. 4)	6,768	13,540	68.2	-
	717	1,438	68.2	0.7
corvette5	11,155	21,750	52.3	-
	1,472	2,735	51.9	1.4
isis (Fig. 5)	45,749	91,494	173	-
	5,891	11,778	173	7.6
r96012-250	126,157	250,413	2,942	-
	20,247	38,828	54	29
igea	134,345	268,686	47.14	-
	19,512	39,020	47.14	22.6
total88	876,487	1,812,094	5.98	-
	12,133	33,002	41.9	76

Table 2: Statistics relative to geometry-driven mesh simplification (option `-O -2`).

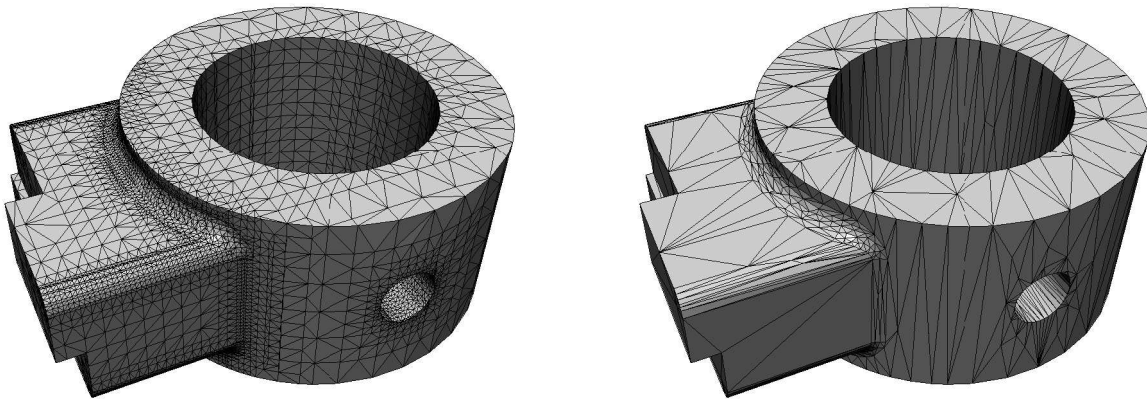


Figure 4: Example of geometry-driven mesh simplification using option `-O -2`. Left : original mesh. Right : simplified surface mesh.

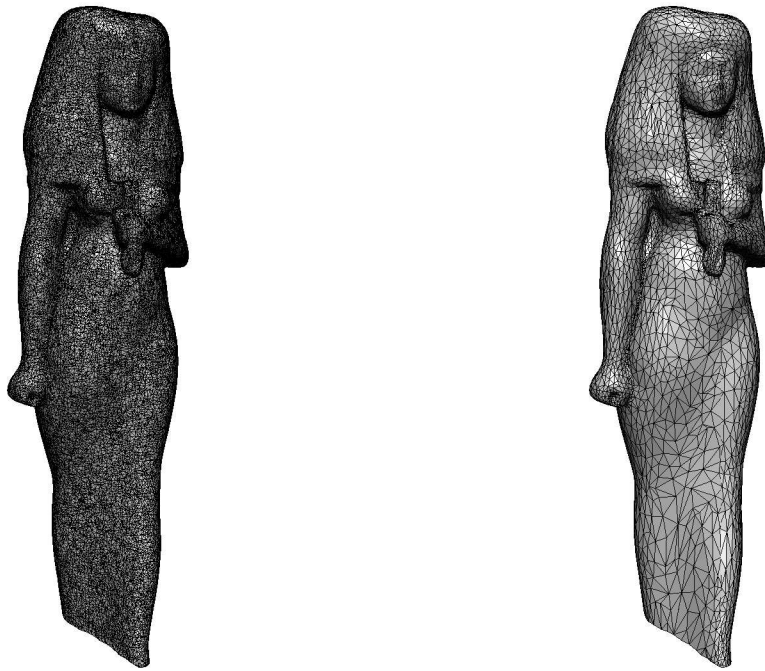


Figure 5: *Example of geometry-driven mesh simplification using option -O -2. Left : original mesh. Right : simplified surface mesh.*

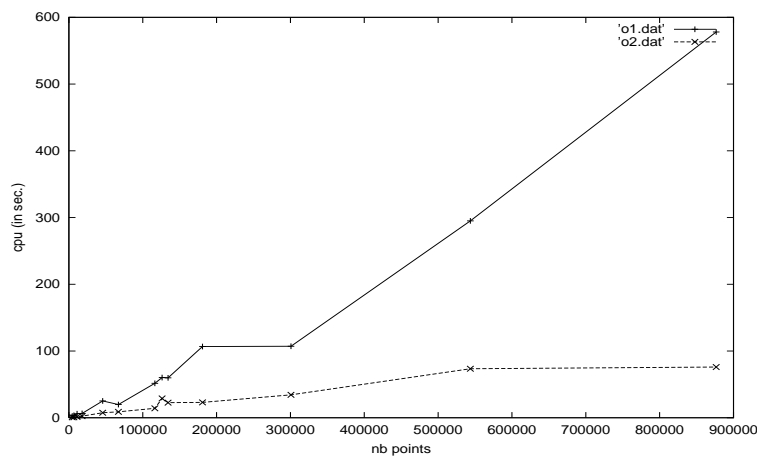


Table 3: *Evolution of cpu time (in sec. on a HP 9000 workstation) vs. number of points.*

Table 3 shows the evolution of the cpu time for various surface meshes for both decimation options (-O -1 and -O -2). In this table, the user can see that option -O -2 runs always faster than option -O -1. This plot tends to confirm the intuition that extracting a high-quality mesh requires more work than simply creating a geometric mesh.

### 6.1.3 Application : creation of hierarchical meshes

In some cases, for instance for visualization purposes, it may be desirable to create various occurrences of the same model at different level of detail. This is possible using YAMS with options `-O -1` or `-O -2` increasing the value of the `Tolerance` parameter (see example Figure 6). It is also possible to generate a surface mesh at any level that is composed of a subset of the nodes of the initial triangulation. In this case, the user has to specify `-ns` on the command line to prevent the nodes to be relocated during the simplification stage. The initial mesh (not displayed in Figure 6) contains 543,652 vertices.

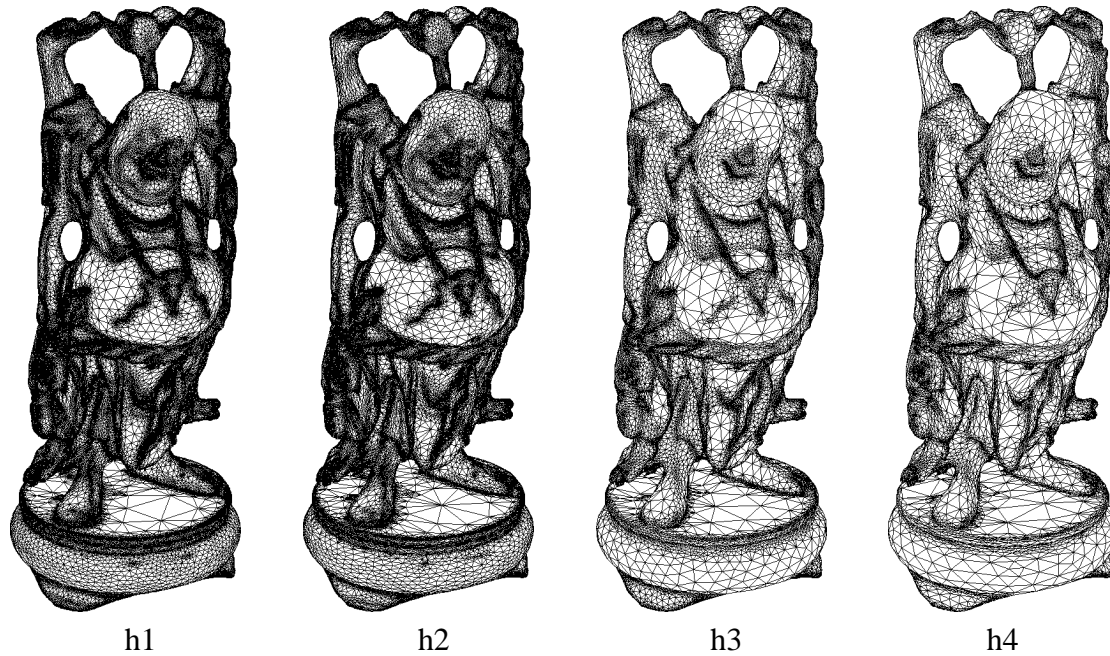


Figure 6: *Geometric mesh simplification applied to the construction of hierarchical meshes. The meshes contains 216,861 vertices, 166,232 vertices, 63,399 vertices and 43,268 vertices, respectively (data courtesy Stanford University, CG dept.).*

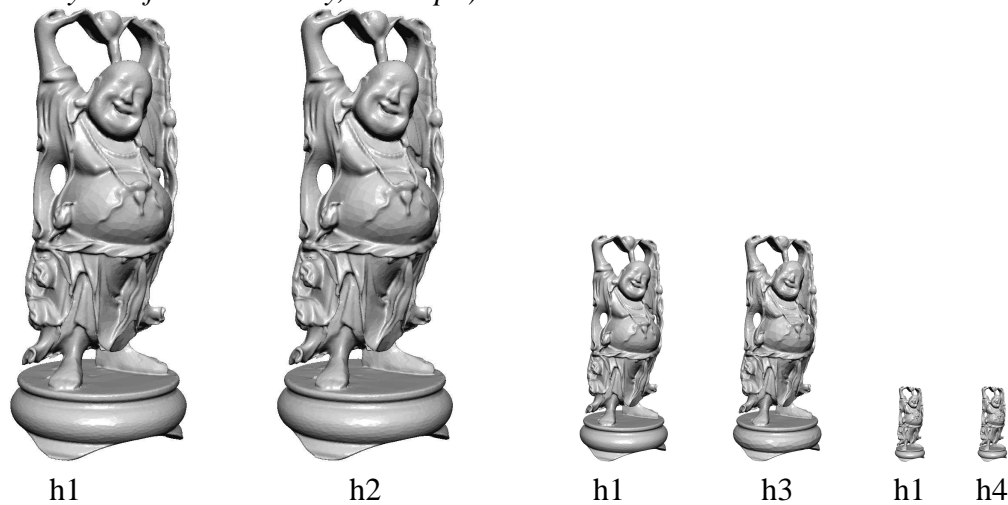


Figure 7: *Use of hierarchical meshes for visualization purposes.*

## 6.2 Surface mesh enrichment

If the given surface triangulation is a poor approximation of the underlying surface geometry or if it does not have enough nodes or if the mesh elements are not well shaped, then surface mesh enrichment becomes necessary. In this context, YAMS offers the possibility of enriching a given mesh, *i.e.* of adding vertices on the surface. The software being not directly related to any CAD system, a procedure has to be defined to answer a query like : "given a point find the closest point onto the surface". To this end, a geometric support ( $G^1$  continuous) is defined internally that will be used to locate points onto the surface and to answer queries, hence emulating the behavior of a simple geometric modeller. This geometric support represents the analytical definition of the underlying geometry [6, 7].

Again, two options are concerned with this feature, that allow to define :

- high-quality surface meshes (for numerical simulations),
- geometric surface meshes (without element shape quality control).

### 6.2.1 High-quality geometric surface meshes

This feature corresponds to option `-O 1` of the command line. As for the corresponding decimation option, an internal curvature-based metric is constructed that prescribe, at each mesh vertex, the desired element size, depending on the `Tolerance`, `GeomApp` and `Gradation` values specified.

Table 4 reports statistics about the mesh enrichment procedure for various surface meshes. In this table,  $np$  and  $ne$  represent the number of points and triangles,  $Q_{max}$  and  $Q_{avg}$  represent respectively the worst and average element shape quality,  $Q_{[12]}$  is the percentage of elements having a quality better than 2,  $l_{avg}$  and  $\tau$  represent the average edge lengths and the efficiency coefficient,  $cpu$  denotes the CPU time (on a HP 9000 workstation). In this table, the first line of each example indicates the initial values and the second line the final results.

<i>Test cases</i>	$np$	$ne$	$Q_{max}$	$Q_{avg}$	$Q_{[12]}(\%)$	$l_{avg}$	$\tau$	$cpu$ (sec.)
coupelle	329	540	17.4	2.1	74.8			-
	1,555	2,778	10.1	1.5	84.	0.91	0.94	1.7
pump (Fig. 1)	1,646	3,312	45.7	5.2	32.5			-
	7,101	14,222	11.7	1.4	88.8	0.92	0.94	11
thepart (Fig. 8, top)	994	1,992	65	7.1	18.4			-
	7,377	14,758	13.6	1.3	97.7	0.95	0.95	16.2
b1	3,013	6,062	4.3	1.4	89.6			-
	18,535	37,106	8.9	1.4	89.4	0.52	0.7	41.1
airbus	4,878	9,752	8.5	1.1	99.1			-
	17,608	35,212	16.4	1.4	96.7	0.8	0.89	29.
modelb	17,445	34,930	9.5	1.4	98.39			-
	21,500	43,040	14	1.3	94.76	0.9	0.93	41

Table 4: Statistics relative to high-quality mesh enrichment (option `-O 1`).

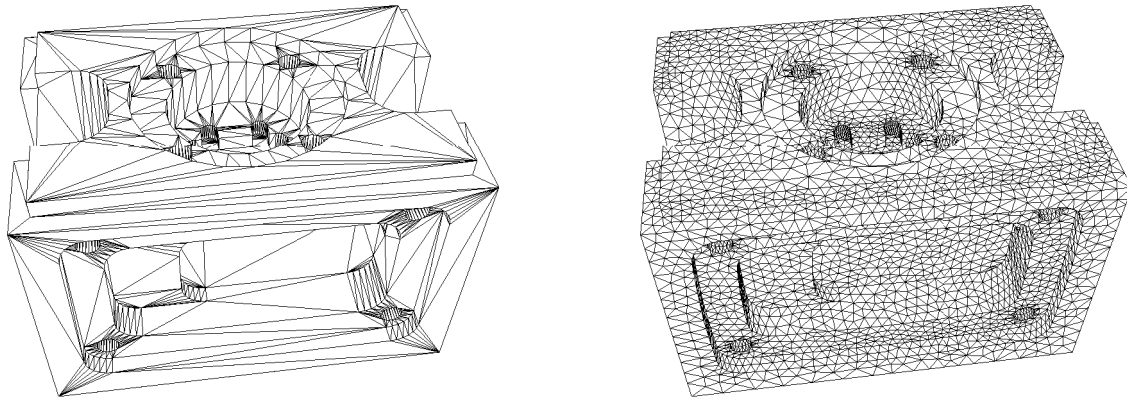


Figure 8: *Example of high-quality surface mesh enrichment. Left : original surface triangulation. Right : high-quality mesh.*

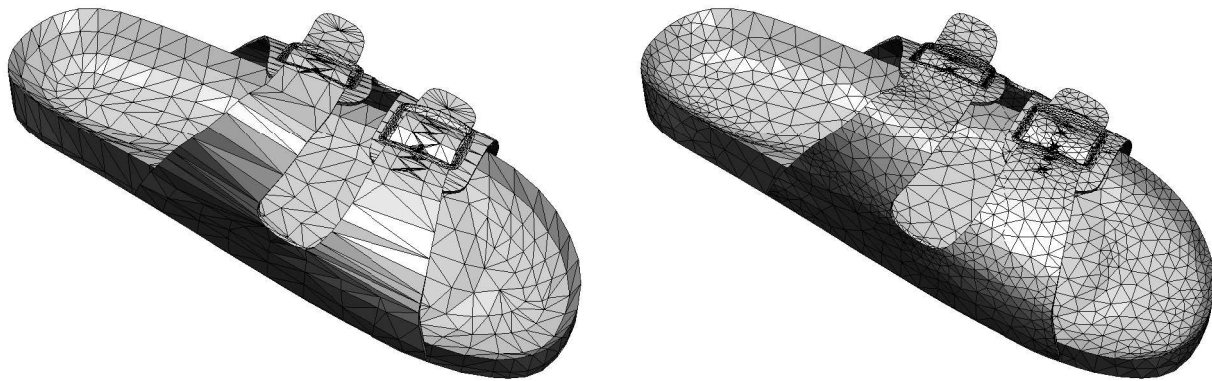


Figure 9: *Example of high-quality surface mesh enrichment. Left : original surface triangulation with minimal number of vertices. Right : high-quality mesh.*

### 6.2.2 Geometric surface meshes

This feature correspond to option `-O 2` of the command line. As for the corresponding decimation option, the resulting surface meshes are slightly anisotropic. Here also, the meshes are primarily intended for visualization purposes, when the element shape quality is less important than preserving the geometric accuracy of the surface discretization.

**Remark 6.1** *In this version, the option `-O 3` of the command line allows the user to stop after the point insertion stage (without removing extra vertices). This makes possible the creation of geometrically accurate meshes that may be considered as geometrical support (i.e., the mesh stands for the geometry definition, cf. Figure 10).*

### 6.2.3 Mesh gradation control

The element shape quality is of utmost importance in numerical simulations as it may impact the accuracy of the numerical solutions. Usually, the quality of an element can be controlled by the surface remeshing program. However, the shape quality is also related to the local mesh density and the size distribution function [2]. YAMS allows the user to control the rate of the size variation

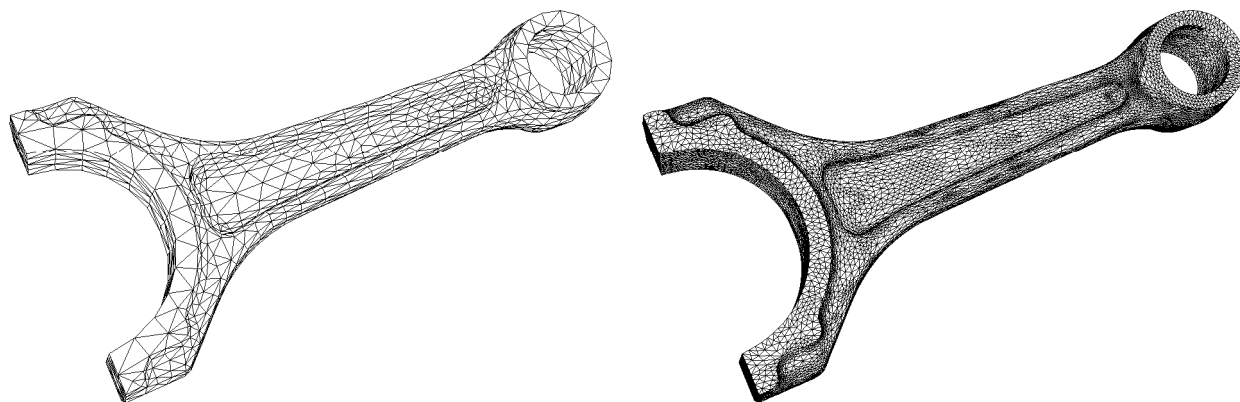


Figure 10: *Example of the creation of a geometric surface mesh using option `-O 3`.*

(i.e., how the element size changes along a mesh edge), by adjusting the parameter `Gradation`. Figure 11 illustrates the influence of the mesh gradation parameter.

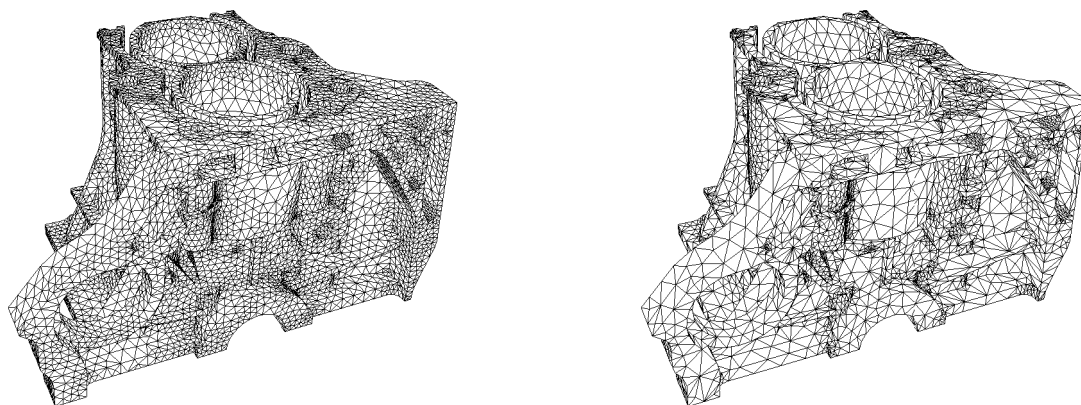


Figure 11: *Example of mesh gradation control. Left : Gradation parameter set to 1.3 (default). Right : Gradation parameter set to 4 (data courtesy Honda motor, Japan).*

## 6.3 Surface mesh optimization

As the accuracy of the numerical solutions are related to the mesh quality in finite element/volume methods [3], it may be desirable to improve the mesh quality, while keeping the number of nodes unchanged. This requirement is also of interest as the quality of a surface mesh (defining the boundary of a volume) affects the quality of a 3D tetrahedral mesh.

In this respect, YAMS allows one to improve the element shape quality of the given surface triangulation, while preserving the geometric approximation.

### 6.3.1 Mesh quality improvement

A straightforward way of optimizing a mesh is to keep the number of mesh vertices constant and simply make local adjustments (edge flipping, node relocation) so as to improve the element shape quality. This feature corresponds to option `-O 0`. Figure 12 illustrates the quality improvement procedure.

**Remark 6.2** *This option may also be useful to slightly improve the quality of a surface mesh processed by YAMS. It usually leads to remove badly-shaped elements that a previous treatment was unable to delete (as local curvatures are recomputed).*

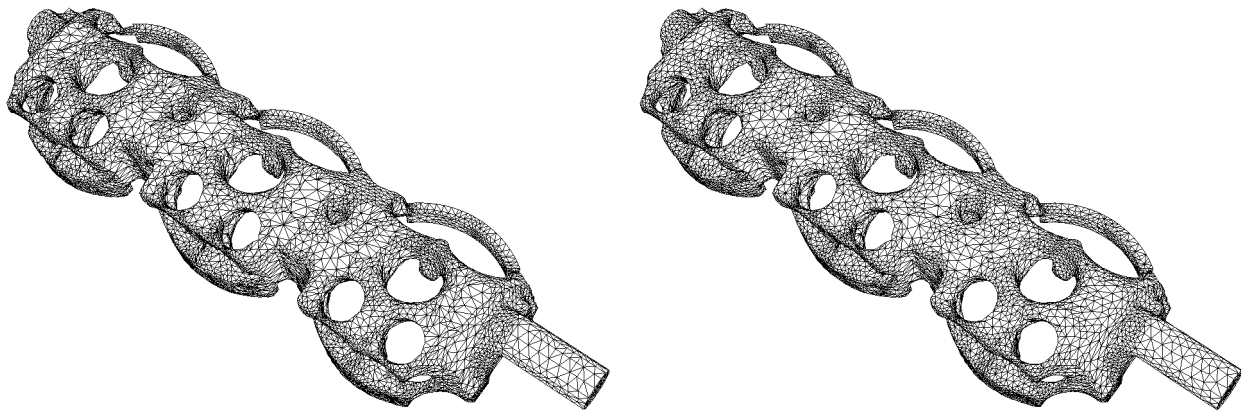


Figure 12: *Example of surface mesh optimization. Left : original surface triangulation (data courtesy IFP, Paris). Right : optimized surface mesh.*

### 6.3.2 Surface approximation improvement

The "Marching Cubes" algorithm is one of the most popular surface reconstruction algorithm when dealing with discrete data (for instance, supplied by sensing or scanning devices). However, the major drawbacks of this algorithm is that :

- i) it usually produces too many mesh elements,
- ii) the surface approximation is usually very poor (staircase effect).

YAMS provides an easy solution to overcome these problems. Using option `-O 9` the code can improve the surface approximation (*i.e.*, remove the staircases). Then, the user may choose to simplify the mesh using options `-O -1` or `-O -2`. Figure 13 illustrates the option `-O 9` on a biomedical data.



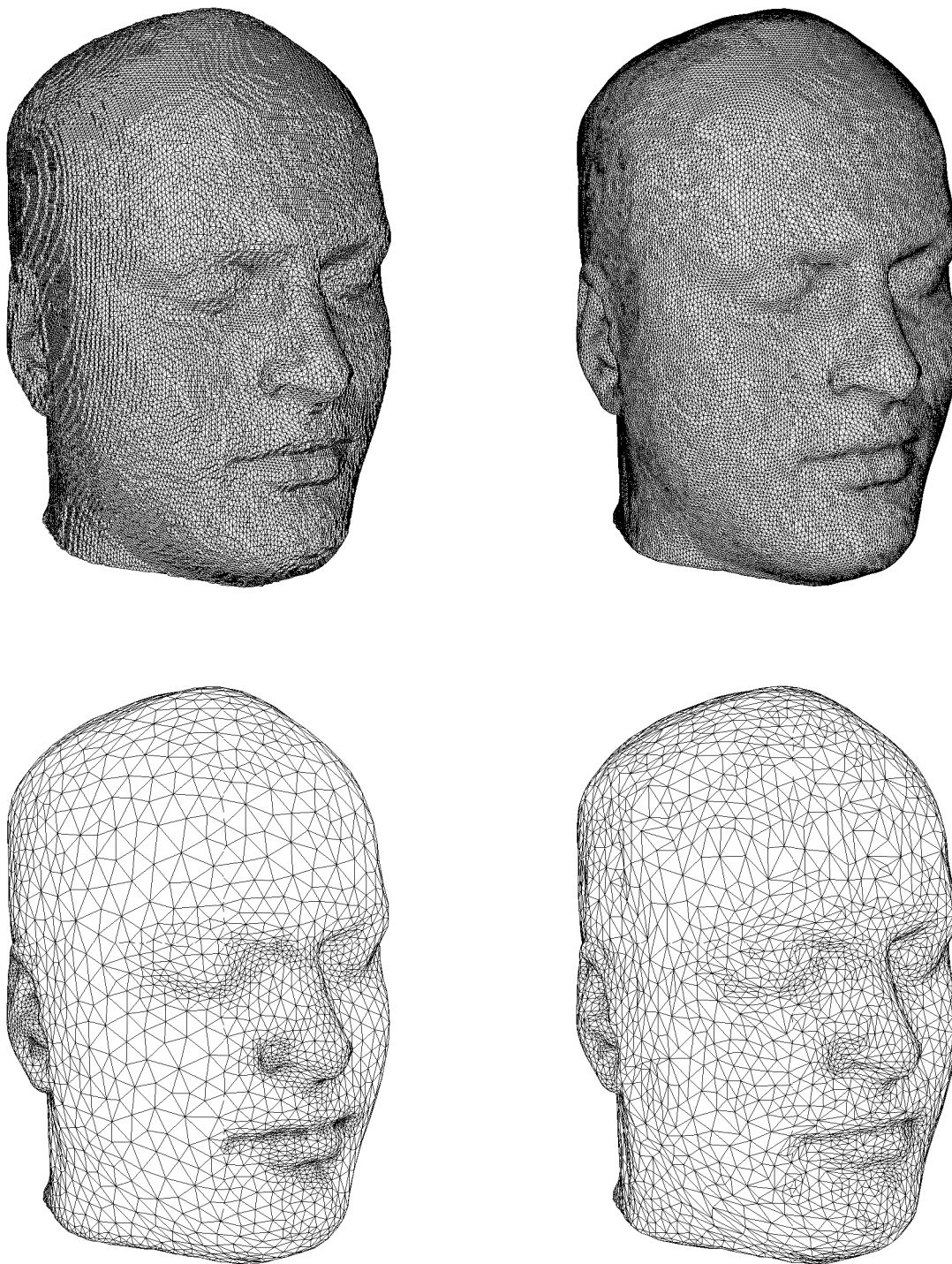


Figure 13: *Example of surface approximation improvement. Top left : original mesh (67,108 vertices), top right : 'smoothed' mesh after 100 iterations. Bottom left : after option -O -1 (9,013 vertices, 20.1 sec), bottom right : after option -O -2 (5,883 vertices, 9.7 sec.).*

## 6.4 Mesh adaptation

In the context of numerical simulations based on finite element/volumes, it is often desirable to control the mesh density more precisely than just taking the surface geometry into account. For instance, in CFD, it is required to have small elements in shock-wave or boundary layer regions, even if the geometry is locally planar. The mesh density is then prescribed using a user-specified size map (for instance constructed from an *a posteriori* error estimate).

In this regard, YAMS can also be used as a mesh adaptation tool, to adapt the mesh to the problem to solve (*i.e.*, to a specific size map). To this end, the algorithm will combine the geometric metric (computed internally) with the user supplied metric. Hence, the resulting mesh will have a mesh density related to the local curvature of the surface as well as to the user prescribed size map. This feature can be achieved using the option `-O 1` (the one used to create high-quality meshes) in combination with the reading of a `xxx.bb` file to feed the algorithm with a required size per vertex.

Figure 14 shows an example of mesh adaptation on a CFD example (transonic flow over a ONERA M6 wing, Mach 0.84, angle of attack 3.06).

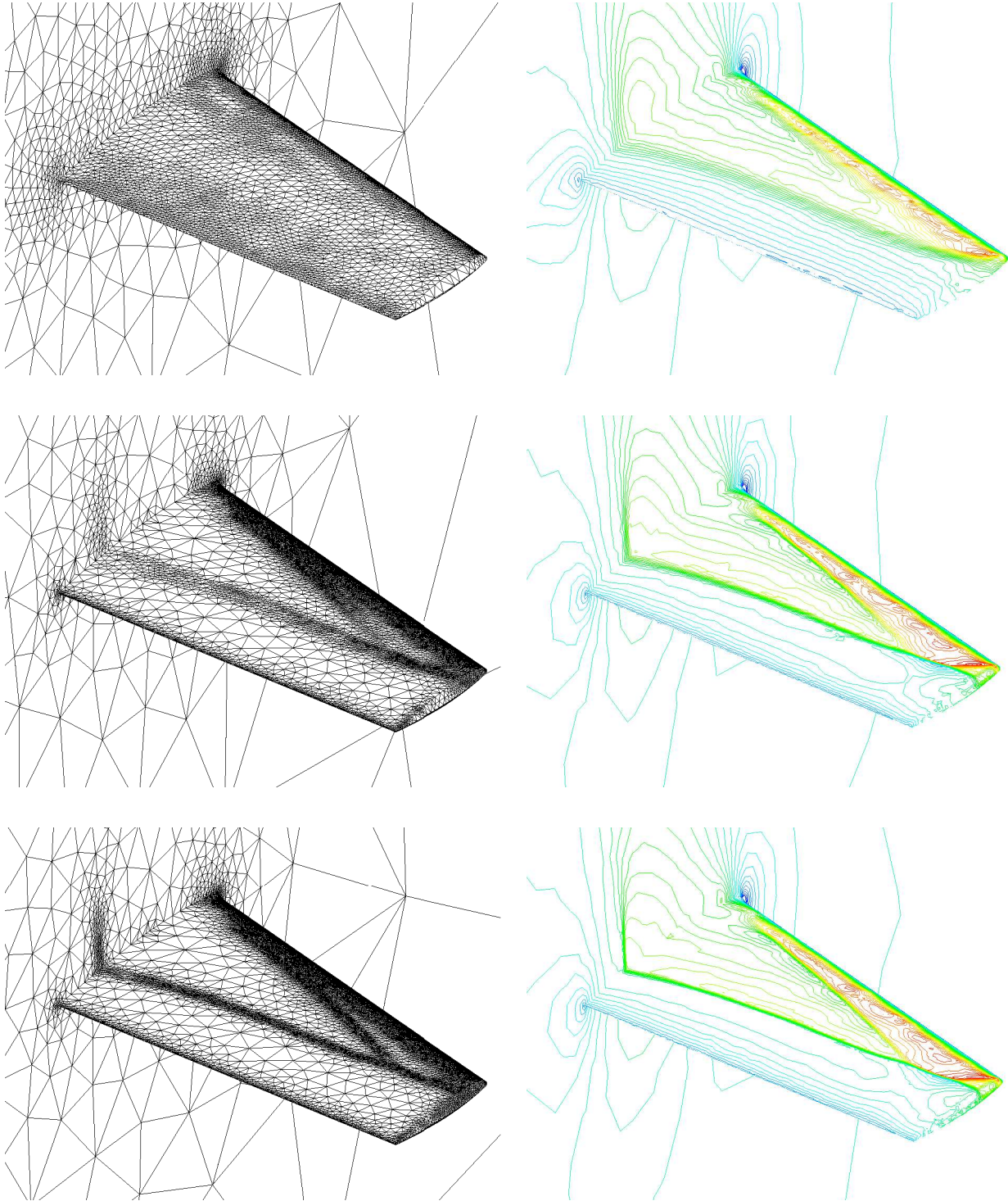


Figure 14: *ONERA M6 wing : adapted meshes and iso-mach lines associated to iterations 1 (24,039 vertices), 4 (96,319 vertices) and 7 (120,343 vertices).*

## 6.5 2D applications

Although YAMS is inherently a surface remeshing software, it can also process 2D (triangular) meshes. In this case, 2D triangulations are considered as planar surfaces. All available options (except option -O 9) apply to planar meshes as well as to surface meshes. Figures 15 and 16 illustrate the optimization and adaptation of planar meshes.

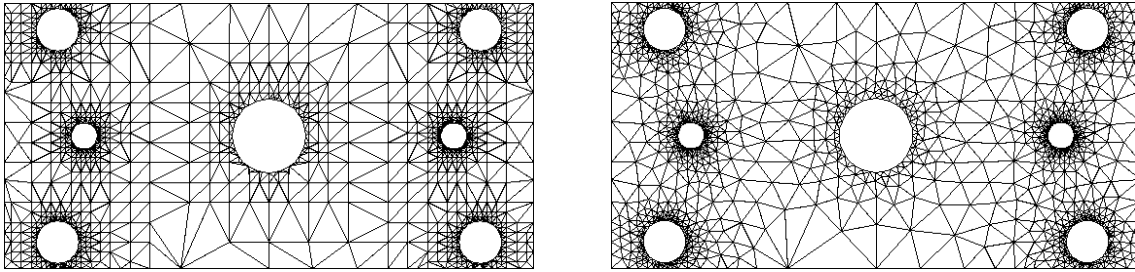


Figure 15: *Example of planar surface remeshing.*

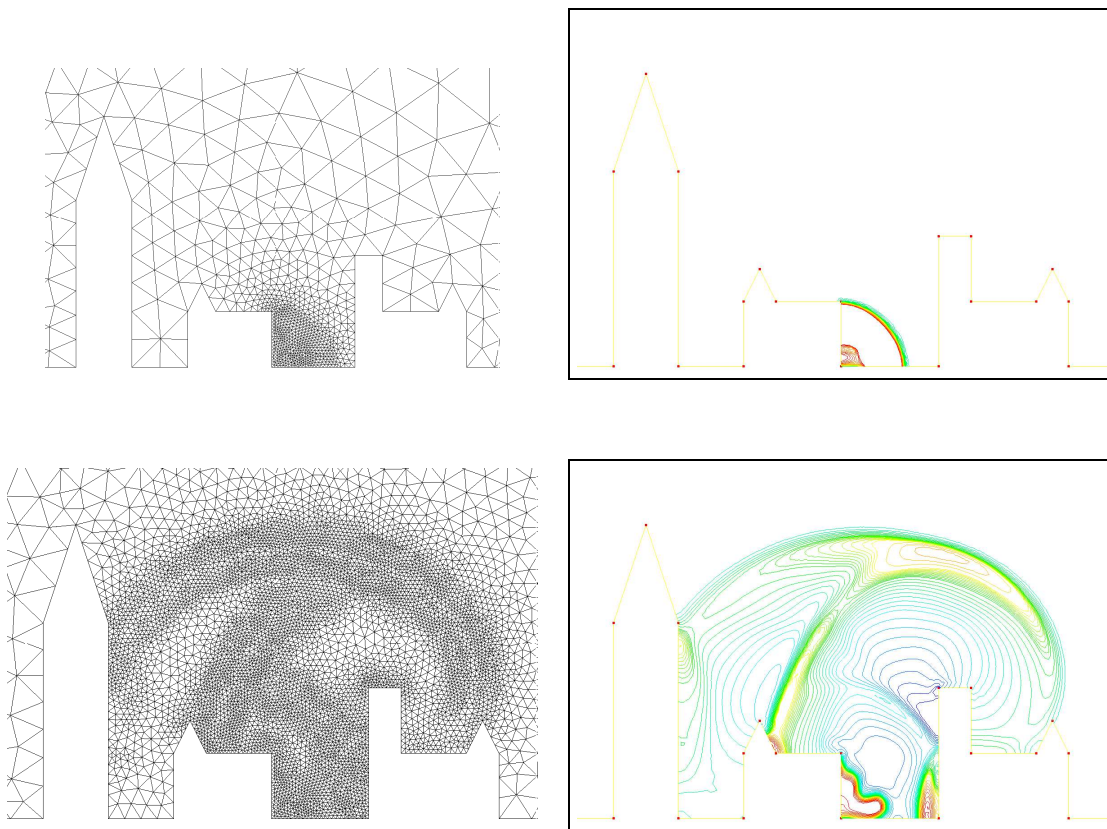


Figure 16: *Example of planar surface adaptation in CFD. Left : original (878 vertices) and final adapted (6,424 vertices) meshes. Right : the corresponding isodensity values.*

## 7 Limitations

This section is intended for making precise several assumptions about surface meshing. It consists of a series of answers about surface remeshing questions the users may be faced with. It is also intended for precisising the limits of the software (*i.e.*, the things it cannot do)<sup>3</sup>.

### 7.1 Defining the surface geometry

As already mentioned, YAMS is not related to any CAD modelling system. Therefore, the surface geometry is only guessed from the given surface triangulation. To be able to remesh this surface and obtain a high-quality surface mesh, the user must take care of supplying a triangulation that is already a good (if not accurate) piecewise linear approximation of the model geometry. This may not be always possible, especially when dealing with scanned data (see FAQ in Appendix).

Notice that YAMS keeps the topology of the mesh unchanged, if the initial triangulation is not conformal, the resulting mesh will also not be.

Usually, CAD systems are able (*i.e.*, have an option) to output an initial surface triangulation that contains a large numbers of equally spaced nodes thus representing a good description of the underlying surface. In such cases, the task is made easy for YAMS. Indeed, mesh simplification will be more likely to give the desired result.

However, recent experiences have shown that CAD systems now export, presumably optimal, triangulations with a minimal amount of nodes (see Figure 17, for instance). If these triangulations may be used for graphic purposes, they are **not**, by all means, geometric meshes (see Figure 17, right-hand side, for a local enlargement showing the problem). The user must be convinced that YAMS cannot do a good job on such surfaces, because the geometry of the surface is not well captured (the local curvature is violated, the geometric tolerance is varying a lot, etc.). Consequently, YAMS is probably not the right tool to be applied on such triangulations.

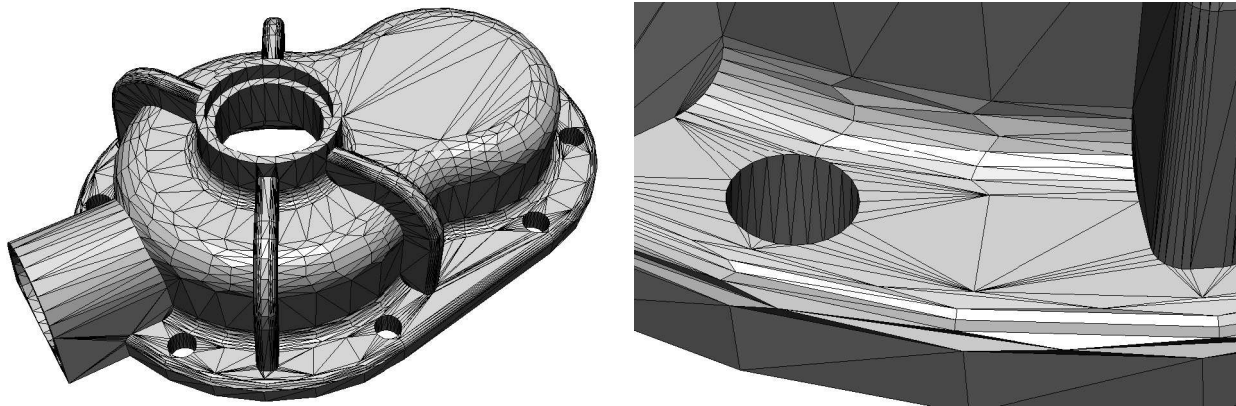


Figure 17: *Example of 'pseudo-geometric' surface triangulation. Right : enlargement on a problem area where the two directions of curvature are not well captured.*

<sup>3</sup>Some of these questions could have figured in the FAQ list.



## 7.2 CAD repair tool

Be conscious that YAMS is not a CAD repair tool. Indeed, the software will preserve the mesh topology and geometry defined by the initial surface triangulation. In particular, if the given triangulation is non-manifold, the resulting mesh will also be non-manifold (although option `-cn` can sometimes get rid of the problem by saving a specific connected component). Similarly, YAMS will preserve the mesh conformity and will not fill the holes and gap between elements. If the surface is self-intersecting, this characteristic will also remain in the final mesh.

## 7.3 About corners and ridges

Usually, corners and ridges are defined based on the `Ridge` parameter in the `.yams` file. Ridges are defined when the dihedral angle between 2 faces exceeds the prescribed limit. Corners are defined when 3 incident ridges meet at a vertex or if 2 ridges form a small angle.

The limit between 2 sub-domains (neighboring triangles with different references) defines a curve traced onto the surface. This curve will be remeshed according to the prescribed tolerance value, thus inducing some more constraint in the remshing process.

## 8 Appendix

This section will provide information about the error messages, file formats, shape and size quality functions used in YAMS software. At the end of this section a list of Frequently Asked Questions will help the user to better understand the main concepts and control parameters of YAMS.

### 8.1 List of error messages

Warnings, errors and possible cause of failure are identified and listed hereafter. Usually the syntax of error messages is the following :

ERR *xxxx*, *proc*, MESSAGE, *list*,

where *xxxx* stands for the error number, *proc* is the name of the procedure in which the error was detected, MESSAGE is the error diagnostic and *list* is a (facultative) list of arguments related to this error (e.g. a list of mesh entities). The first digit of the error number indicates the stage of the process in which the error occurred :

- 0 the process was in the preliminary stage (reading input files),
- 1 analysis stage : surface orientation, normals and tangents computation, etc.,
- 4 surface remeshing problems,
- 9 runtime or system-dependent errors : cancelled job, hardware problem, etc.

The list of diagnostics is as follows :

[L0 ] Level 0 : input data related errors.

These errors are mainly related to incorrect data file or memory allocation problems.

ERR	MESSAGE	DIAGNOSTIC / CORRECTION
0000	WRONG DATA TYPE, <i>i</i> .	Incorrect data file (wrong dimension or vertex field missing).
0001	NO INPUT DATA	No element found in the input mesh.
0002	NOT ENOUGH MEMORY	Not enough memory space to store the mesh structure. Solution : use option [ -m MegaBytes ] on the command line to increase mem. size.
0003	WRONG FILE NAME	Check spelling.
0004	DATA NOT CONSIDERED	This data is not processed.
0005	UNABLE TO OPEN FILE	Unable to open data file. Solution : set the correct read permissions.
0010	NORMAL AT VERTEX DISCARDED	Normal vector discarded (wrong vertex number)
0011	NORMAL AT TRIANGLE VERTEX DISCARDED	Normal vector discarded
0020	FACE DISCARDED	Face <i>i</i> discarded from the triangulation.
0021	EDGE(S) IGNORED	Set of edges discarded (memory problem).
0022	<i>n</i> FACES DISCARDED	Number of discarded faces
0030-33	MEMORY REALLOC PROBLEM (HASH, NORMALS, TANGENTS)	Unable to expand memory space. Solution : use option [ -m <i>n</i> ]

[L1 ] Level 1 : Surface analysis and topology errors.

These errors are mainly related to topology problems (e.g., orientation, etc.).

ERR	MESSAGE	DIAGNOSTIC / CORRECTION
1001	EDGE PROBLEM $(i, j) : k$	Edge $(i, j)$ , face $k$ discarded.
1010	$n$ ADJACENCY PROBLEM	Same face occurs at least twice.
1011	ORIENTATION PROBLEM	Surface mesh is not orientable.
1012	WRONG TOPOLOGY	Mesh topology corrupted.
1013	EDGE FLIP PROBLEM	Unable to update data structure.
1014	WRONG EDGE TOPOLOGY	Unable to update adjacent faces.
1015	EDGE NOT CORRECTLY HASHED	Internal data structure corrupted.
1020	NO ROOT FOR COMPONENT	Unable to correctly identify connected component $i$ .

[L4 ] Level 4 : Surface remeshing errors.

ERR	MESSAGE	DIAGNOSTIC / CORRECTION
4000	UNABLE TO CREATE NEW POINT	Not enough memory to add new vertex.
4001	UNABLE TO CREATE NEW TRIANGLE	Not enough memory to create new triangle.
4006	POOR SHAPE QUALITY	Element $k$ is very badly-shaped.
4007	SURFACE TOO SMALL	
4008	SMALL INRADIUS	Badly shaped element.
4010	CURVE TANGENT PROBLEM	Curve equation not reliable. Solution : use option <code>-nm</code> to avoid building geometric support.
4011	NORMAL VARIATION	New vertex normal differs from old one by more than 15 deg.
4012	INCONSISTENT BALL OF VERTEX	Topology corrupted.

[L8 ] Level 8 : Output errors.

ERR	MESSAGE	DIAGNOSTIC / CORRECTION
8000	UNABLE TO SAVE MESH FILE.	Check write permissions
8001	UNABLE TO SAVE METRIC FILE.	
8003	CURRENT MESH NOT SAVED.	Because of a previous error

[L9 ] Level 9 : errors related to software problems (floating point exceptions, segmentation fault, job cancelled, etc.). The current mesh will not be saved.

ERR	MESSAGE
9900	ABNORMAL STOP.
9901	FLOATING-POINT EXCEPTION.
9902	ILLEGAL INSTRUCTION.
9903	SEGMENTATION FAULT.
9904	PROGRAM KILLED.



## 8.2 Isotropic remeshing

The aim of the surface remeshing algorithm is to produce high-quality geometric surface meshes. This means that the desired element shape is the (nearly) equilateral triangle. The element size is locally related to the surface curvature as prescribed by the geometric size map. The other requirement is to obtain a valid and accurate piecewise linear approximation of the surface geometry. Therefore, the metric map that is constructed internally is able to take into account both the shape and size requirements.

### 8.2.1 Shape quality

For a triangle, the shape quality function measures the aspect ratio of the element and is defined as follows :

$$\mathcal{Q}_K = \alpha \frac{h_{max}}{\rho_K}, \quad (1)$$

where  $h_{max}$  is the element *diameter* (i.e., its longest edge),  $\rho_K$  is the *inradius* of triangle  $K$  and  $\alpha$  is a normalization coefficient chosen so that the quality of the equilateral triangle is equal to 1. Notice that this function ranges between 1 and  $\infty$ , the optimal value being 1. For a mesh  $\mathcal{T}$ , a global quality measure is computed as :

$$\mathcal{Q}_{\mathcal{T}} = \max_{K \in \mathcal{T}} \mathcal{Q}_K.$$

### 8.2.2 Size quality

The length of a mesh edge  $AB$  with respect to the size specification can be computed as :

$$l_{AB} = d(A, B) \int_0^1 \frac{1}{h(t)} dt, \quad (2)$$

where  $d(A, B)$  is the Euclidean distance between  $A$  and  $B$  and  $h(t)$  is a monotonous size variation function along  $AB$ . Furthermore, an efficiency index can then defined as :

$$\tau = 1 - \frac{1}{na} \sum_{i=1}^{na} (1 - e_i)^2, \quad (3)$$

with  $e_i = l_i$  if  $l_i < 1$  and  $e_i = 1/l_i$  if  $l_i > 1$ ,  $na$  being the number of mesh edges. A value of  $\tau$  greater than 0.93 indicates that the resulting mesh conforms well to the size map.

### 8.2.3 Geometric support

To improve the accuracy of the piecewise linear approximation of the surface, a geometric support of class  $G^1$  is locally constructed. It is used to locate a point on the surface when splitting an edge or moving a node. To some extent, this support emulates some queries of a geometric modeller.

### 8.3 File formats

As mentioned in the second section, the geometry of the surface is described using a simple mesh format. The mesh format allows to specify surface features such as ridges, corners, normals, etc.

### 8.4 The mesh format

This format is composed of a single (binary or text) data file. Its structure is organized as a series of fields identified by keywords. The blanks, "newline" or <CR> and tabs are considered as item separators. A comment line starts with the character # and ends at the end of the line. The comments are placed exclusively between the fields.

The mesh file must start with the descriptor :

```
MeshVersionFormatted 1
Dimension 3
```

The other fields supported by YAMS are either required or facultative. The required fields correspond to the geometry (*i.e.*, the coordinates) and to the topology description (*i.e.*, the mesh entities). In the following tables, the term  $v_i$  indicates a vertex number (*i.e.*, the  $i^{th}$  vertex in the vertex list),  $e_i$  is an edge number,  $t_i$  is a triangle number and  $q_i$  is a quadrilateral number. Notice that the vertices are real numbers in single precision.

Keyword	Card.	Syntax	Range
Vertices	$np$	$x_i y_i z_i ref_i$	$\{i = 1, np\}$
Edges	$na$	$a_i^1 a_i^2 ref_i$	$\{i = 1, na\}$
Triangles	$nt$	$v_i^j ref_i$	$\{i = 1, nt\}, \{j = 1, 3\}$
Quadrilaterals	$nq$	$v_i^j ref_i$	$\{i = 1, nq\}, \{j = 1, 4\}$

Then, follows the description of constrained entities or singularities. In particular, a corner point Corner is a point where there is a  $C^0$  continuity between the edges sharing it (this type of item is necessary a mesh vertex). By analogy, a Ridge is an edge where there is a  $C^0$  continuity between the adjacent faces. The fields of type Requiredxx make it possible to specify any type of entity that must be preserved by the meshing algorithm.

Keyword	Card.	Syntax	Range
Corners	$nc$	$v_i$	$\{i = 1, nc\}$
RequiredVertices	$nrV$	$v_i$	$\{i = 1, nrV\}$
Ridges	$nr$	$e_i$	$\{i = 1, nr\}$
RequiredEdges	$nre$	$e_i$	$\{i = 1, nre\}$
RequiredTriangles	$nrt$	$t_i$	$\{i = 1, nrt\}$
RequiredQuadrilaterals	$nrq$	$q_i$	$\{i = 1, nrq\}$

As mentioned above, it is also possible to specify normals and tangents to the surface. The normals (resp. tangents) are given as a list of vectors. The normal at a vertex, keyword NormalAtVertices, is specified using the vertex number and the index of the corresponding normal vector. The normal at a vertex of a triangle, NormalAtTriangleVertices, corresponds to the combination of the triangle number, the index of the vertex in the triangle and the index of the normal vector at this vertex. Similarly for the field corresponding to the keyword NormalAtQuadrilateralVertices. The tangent vectors are described in the same way.

Keyword	Card.	Syntax	Range
Normals	$nn$	$x_i \ y_i \ z_i$	$\{i = 1, nn\}$
Tangents	$nnt$	$x_i \ y_i \ z_i$	$\{i = 1, nnt\}$
NormalAtVertices	$nv$	$v_i \ n_i$	$\{i = 1, nv\}$
NormalAtTriangleVertices	$ntv$	$t_i \ v_j \ n_i$	$\{i = 1, ntv\}$
NormalAtQuadrilateralVertices	$nqv$	$q_i \ v_j \ n_i$	$\{i = 1, nqv\}$
TangentAtEdges	$te$	$e_i \ v_j \ t_i$	$\{i = 1, te\}$

Finally, the data structure must end with the keyword : End.

### 8.4.1 Example

```
# Mesh generated by YAMS (INRIA) V2.1
MeshVersionFormatted 1
Dimension      3

# Set of mesh vertices
Vertices
7902
-3.299989 1.829990 -4.139999 0
-3.209990 1.620000 -3.939990 0
.....

# Set of mesh edges (v1,v2,ref)
Edges
891
401 13 0
13 389 0
410 15 0
.....

# Set of mesh triangles (v1,v2,v3,ref)
Triangles
15346
13 389 401 0
410 15 406 0
.....

# Set of corners
Corners
82
1
2
9      (point 9 is corner)
.....

# Set of ridges
Ridges
891
1
```

```

2
3
.....

# List of normal vectors
Normals
8299
0.244684 0.031587 -0.969088
0.254205 0.033309 -0.966576
.....

# Normals at vertices
NormalAtVertices
7006
406 1391
412 6      (normal vector 6 associated with vertex 412)
.....

End

```

## 8.5 The msh2 format

This file format is a very crude data format provided for backward compatibility purposes. It is composed of two ASCII files, `xxx.points` and `xxx.faces`. The file `.points` has the following structure :

$np$

$x_i \ y_i \ z_i \ ref_i$

representing the vertex coordinates in single precision and the vertex reference. The file `.faces` represents the mesh topology and contains the following records :

$nf$

$d \ v_i^1 \ v_i^2 \ v_i^3 \ ref_i \ ref_i^1 \ ref_i^2 \ ref_i^3$

where  $d$  is either 3 (triangles) or 4 (quad),  $v_i$  represents a vertex number,  $ref_i$  is the reference of the sub-domain containing the face and  $ref_i^j$  are the edge references.

## 8.6 the bb format

This ASCII file contains scalar values associated with a mesh file. It is used to specify the sizes associated with mesh vertices. This file has the following structure :

3 1 1  $np$  2

$s_i, \{j = 1, np\}$

The values  $s_i$  represent the local sizes associated with mesh vertices. Notice that the values 3 1 1 and 2 on the first record must be specified as it (other codes use different values).

## 8.7 List of FAQ's

The aim of this section is to provide a set of questions/answers that can help the user to obtain a correct and accurate surface mesh from a given surface triangulation. In some cases, the answers are very short, while for others, the answer is more elaborate. In any case, the authors encourage the user to play with the various options and parameters to get a better understanding of YAMS behavior.

Q1. *How to get a high-quality mesh suitable for FE computations ?*

Option `-O 1` will do the job. Also use option `-O 0` on a given mesh to improve the mesh quality. Notice also that the mesh quality is related to the `Gradation` parameter value (a value of 1.3 should give good results).

Q2. *How to get an optimal mesh ?*

Usually, the resulting mesh is a good compromise between the quality of the geometric approximation, the element shape quality and the number of elements. In this sense, the mesh is 'optimal'.

Notice that this mesh is not **Delaunay** ! (as the Delaunay property needs to be defined on surfaces...).

Q3. *How to control the number of nodes/elements ?*

It is not possible to control precisely the number of vertices in a mesh. However, running YAMS two or three times and adjusting slightly the `Tolerance` or `GeomApp` parameters, you should be able to get a mesh having roughly the desired number of vertices.

Q4. *How to prevent the vertex positions to be modified ?*

Use `yams -ns` to prevent the mesh vertices to be changed.

Q5. *How to preserve specific mesh entities ?*

In a mesh file Use the fields `Requiredxx` to constrain mesh entities in a mesh file.

Q6. *How to deal with scanned data ?*

After Marching Cubes reconstruction algorithms, first use option `-O 9` to remove staircases effects. Then, on a second run, use options `-O 1` or `-O 2` to create a high-quality mesh or a geometric mesh.

Q7. *How to remove small features ?*

Set the parameter `MinSize` in the `.yams` file to a suitable value. Notice that this value is used to truncate the intrinsic metric. Hence, if the value is too high, the geometry of the surface can be affected. Another option is to increase the `Tolerance` value and use option `-O 2` to remesh the surface within this tolerance (the tolerance value has to be larger than the size of the small feature to be removed).

Q8. *How to control size variations ?*

The mesh gradation is controlled by the `Gradation` parameter.

Q9. *How to avoid local over-refinement ?*

Usually, if the mesh density is locally too large, this is mainly due to small feature (or highly-curved regions) that may be artifacts. To prevent the creation of too small elements, set the parameter `MinSize` in the `.yams` file to a suitable value.

Q10. *How to remove badly-shaped triangles ?*

Sometimes, poor-shaped triangles remain after surface remeshing. This may be due to geometric constraints (2 ridges too close) or because of numerical roundoff problems. Try to run YAMS again with options `-O 0` or `-O 1` on the resulting mesh to improve the mesh quality.

Q11. *How to create constant size meshes ?*

There are two ways of achieving constant-size meshes. The first one is to set the `MinSize` and `MaxSize` parameters to the same value in the `.yams` file. The other way is to set the `Gradation` parameter to 1 in this file. In the second case, the size of the element will be almost constant, but adjusted to the smallest size in the mesh (dictated by the geometry requirements).

Q12. *How to visualize resulting surface meshes ?*

As other meshing tools developed at INRIA, YAMS does not offer a graphical user interface (GUI). However, a `.mesh` file is easy to read and can thus be send out to standard graphic software packages.

Notice that a natural companion of YAMS is `Medit`, a 3D interactive mesh display software (also developed at INRIA-Rocquencourt). For more information on this software, see :

<http://www-rocq.inria.fr/gamma/medit/medit.html>

## References

- [1] F. ALAUZET *et al.* (2001), Transient fixed point based unstructured mesh adaptation, *ECCOMAS Computational Fluid Dynamic Conf.*, Swansea, UK.
- [2] H. BOROUCAKI, F. HECHT AND P.J. FREY (1998), Mesh gradation control, *Int. J. Numer. Methods Eng.*, **43**(6), 1143-1157.
- [3] P.G. CIARLET (1991), Basic Error Estimates for Elliptic Problems, in Handbook of Numerical Analysis, vol II, Finite Element methods (Part 1), P.G. Ciarlet and J.L. Lions Eds, North Holland, 17-352.
- [4] P.J. FREY AND P.L. GEORGE (2000), Mesh generation. application to finite elements, Hermès Science Publ., Paris, Oxford, 814 pages.
- [5] P.J. FREY AND H. BOROUCAKI (1998), Geometric evaluation of finite element surface meshes, *Finite Elements in Analysis and Design*, **31**, 33-53.
- [6] P.J. FREY AND H. BOROUCAKI (1998), Geometric Surface Mesh Optimization, *Computing and Visualization in Science*, **1**, 113-121.
- [7] P.J. FREY (2000), About surface remeshing, In *Proc.of 9th Int. Meshing Roundtable*, New Orleans, LO, USA, oct.



---

Unité de recherche INRIA Lorraine, Technopôle de Nancy-Brabois, Campus scientifique,  
615 rue du Jardin Botanique, BP 101, 54600 VILLERS LÈS NANCY  
Unité de recherche INRIA Rennes, Irisa, Campus universitaire de Beaulieu, 35042 RENNES Cedex  
Unité de recherche INRIA Rhône-Alpes, 655, avenue de l'Europe, 38330 MONTBONNOT ST MARTIN  
Unité de recherche INRIA Rocquencourt, Domaine de Voluceau, Rocquencourt, BP 105, 78153 LE CHESNAY Cedex  
Unité de recherche INRIA Sophia-Antipolis, 2004 route des Lucioles, BP 93, 06902 SOPHIA-ANTIPOLIS Cedex

---

Éditeur  
INRIA, Domaine de Voluceau, Rocquencourt, BP 105, 78153 LE CHESNAY Cedex (France)  
<http://www.inria.fr>  
ISSN 0249-6399